International Journal of Mechanical and Production Engineering Research and Development IJMPERD)
ISSN (P): 2249-6890; ISSN (E): 2249-8001
Vol. 9, Issue 4, Aug 2019, 753-760

DESIGN OF CENTRIFUGAL PUMP IMPELLER AND ITS EFFECTS ON CAVITATION

CH. SAI TEJA[1] & Dr. M. MURALI KRISHNA[2]

[1]Student, Department of Aeronautical Engineering, Aerospace, Mallareddy College of Engineering and Technology, Telangana, India

[2]Professor, Department of Aeronautical Engineering, Mallareddy College of Engineering and Technology, Telangana, India

ABSTRACT

A centrifugal pump's function is to transport fluid from one system to another system, within minimum time. Impeller is a key component in centrifugal pump, it intakes fluid through hub of impeller axially and distributes the fluid, radially. The configuration of the impeller blades directly effects the performance. The current paper investigates the characteristics of the pump by altering the blade profile and exit blade angles. A destructive phenomenon of cavitation is greatly influenced by blade profile. Generally, the blades of impeller will have a steep fall from hub to tip or remain in same height from hub to tip. The present project investigates the performance by employing a volute rim connecting all highest points of the blade tips. The blades also rise from tub to tip allowing the fluid to have enough space to accumulate large volumes of fluid with minimal impact on the blades to hub. Any alteration in the geometry, blade profile has a significant influence on the performance of the pump. Npsh denotes the net pressure suction head that is the amount of energy at the pump suction available to exert pressure on the fluid, if the pressure required at the pump inlet to make the liquids flow through suction side without cavitation. This can greatly reduce the cavitation effects, helping the pump to sustain to the calculated service life. Simulation is carried out using ANSYS R 15 software in FLUENT module, with ICEM meshing and SST k-Ω turbulence model.

KEYWORDS: Impeller, Cavitation & Simulation

Received: May 06, 2019; **Accepted**: Jun 17, 2019; **Published**: Jul 10, 2019; **Paper Id**.: IJMPERDAUG201975

Original Article

INTRODUCTION

The Fluid entering a centrifugal pump travels towards the low pressure area that is the impeller eye. Impeller and blades rotate and transfer the momentum to increasing fluid. Fluid of high kinetic energy is forced out of the impeller areas and it enters the casing. The centrifugal pumps draws fluid through the hub axially and transfers the fluid radially, it is driven by a shaft mechanically connected to a motor. It generally can handle large volumes of liquids with a high mass flow rates. Centrifugal pumps can be employed for sewage drainage system, petro chemical industry mining, assisting in fire protection, supplying large volumes of fluid, pumping crude oil, irrigation, pumping the water and other domestic purposes. Centrifugal pump faces the practical problems such as cavitation, which will have detrimental effects such as erosion of the blades which further gives rise to additional problems such as vibrations, ultimately leading to reduction in service life than estimated. Other effects include loss in efficiency and performance of the centrifugal pump. If the absolute static pressure is above the vapor pressure of the liquid in the pump, it is net pressure suction head available and is a function of the system in the pump in which it operates. It is the excess pressure of the fluid over its vapor pressure, as it arrives at the pump

suction. The energy that is available to the liquid at the eye to perform the energy transformation process and assure that the liquid remains in liquid state. The major problem facing by centrifugal pump is cavitation. The present project aims to design and analyze an impeller, which will minimize the cavitation effect of the component. The work includes modeling of the component using catia v5 software, and the analysis carried out by fluent module in ANSYS software, where the flow simulation is carried out.

CAVITATION

The Cavitation is a low pressure region that falls below atmospheric pressure, but the pressure cannot fall below the vapor pressure at a given temperature. If the vapor pressure is reached, the fluid forms small bubbles of vapor in large numbers, carried along with the flow, when it reaches a pressure higher; it suddenly collapses moving from all directions and collides at the centre of the cavity, and giving rise to very high pressure. This process continues several thousand times a minute. Reduced pressure and greater flow velocity with the increased temperature may cause the liquid to flash to stream. Vapor bubbles travel according with. As the velocity of fluid decreases, the fluid pressure increases, the vapor bubbles suddenly collapses on the outer portion of the impeller which can cause a very serious problems for centrifugal pumps. Collapsing voids that implode near to a metal surface cause cyclic stress through repeated implosion, crating a high energy shock wave inside the fluid. This phenomenon of cavitation is destructive in nature.

Types of Cavitation

- **Travelling Cavitation:** It is composed of individual transit cavities and moves with the fluid, may appear at the low pressure points along the solid boundary or in high turbulence region.
- **Fixed Cavitation:** It is developed after the inception, and involves detachment of fluid flow from the rigid boundary of an immersed body to form a cavity attached to the boundary, cavities may have high turbulence boiling surface or it may have smooth surface.
- **Vortex Cavitation:** Cavities are found in the core of vortices, it occurs on the tip of the impeller blades forming in zones of high shear regions, it is also known as tip cavitation.
- **Vibratory Cavitation:** This cavity is formed due to continuous series of high frequency pressure, high amplitude generated by an immersed body, which vibrates continuously.

Figure 1: Cavitation of Impeller

Figure 2: Damaged Impeller

IMPELLER DESIGN

A Three-dimensional model of the impeller was created in CATIA V5R20 software by importing the coordinates of the impeller blades and saved in STEP file format. Select front plane and enter into sketcher module, then using a circle command, a circle was created with diameter 274 mm and exit the sketcher module, enter into pad command, extrude the part with a thickness of 5 mm at the center, the part creates a sketch of a circle with diameter 70 mm, then using extruded cut command, remove the material inside the sketch on the front face and create a sketch of the fin. A pipe of diameter 70 mm with a notch is created using similar procedure. Go to start menu and select mechanical design option, and click on the assembly design module, and then go to insert menu and select new component, similarly add another new component. Click on the edit option, in which we find a move selection, and then choose the smart move mode, a dialogue box appears, check on the automatic constraint creation, then both the parts along with the axis appears, select the baseplate and the pipe surface, the parts are assembled. Check whether the part is assembled or not by selecting the manipulating tool from edit option, and move the component. A naca 0012 coordinates are generated and imported into CATIA software modified with the blade, sweep angle is set at 45^0 throughout the airfoil is trimmed, it is assembled to the baseplate, and then using linear rotation command, an additional 7 blades was created. The height of the blades is 62 mm. Another separate part of the volute rim, was created using the circle command in sketcher module with outer diameter 274 mm and inner diameter of 174 mm. Go to start menu and then to mechanical design and select assembly design. Insert menu, and then select new component option, add the baseplate component again, go to insert and then to new component, add the volute rim to the main part, select the axis of the baseplate and the axis of volute rim, it is assembled with the baseplate.

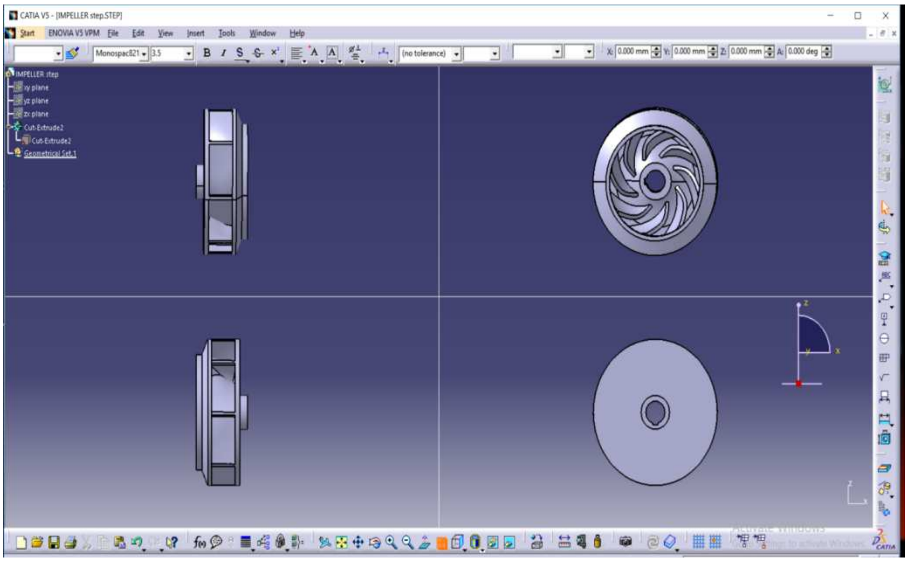

Figure 3: Design of Impeller in CATIA V5 Software

BOUNDARY CONDITIONS

The STEP file was later imported in to software for simulation in ANSYS R 15 workbench- FLUENT, with ICEM meshing SST K-w turbulence model meshing was done on the impeller created. The flow was fully developed while leaving inlet and outlet ducts. The front face of the impeller is taken as inlet, and the rear side is taken as outlet after enclosure from all sides is created on the impeller. Medium of fluid employed for simulation is water with 1000 kg/m^3 density, operating at 25□C temperature. Impeller is rotating at a speed of 1440 rpm (rotations per minute) operating with a motor power of 50 HP. The discharge rate or the fluid flow rate is defined at 400 m^3/hr. The vapour pressure is taken as 24.

Water inlet pipe suction pressure is given as 160mm of hg with water suction head of 20meters. The material used for impeller design is alluminium alloy with material properties of the impeller and casing density is 2.68 gram/ cm^3, modulus of elasticity is 68.3 GPA. In steady state, number of elements employed are 142345, number of nodes taken as 30034. In transient state, number of elements are 466653, number of nodes are 95010.

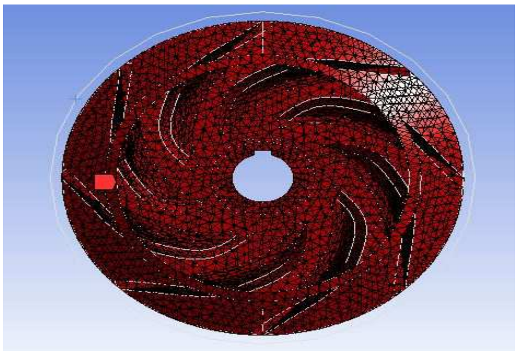

Figure 4: Meshing of Impeller

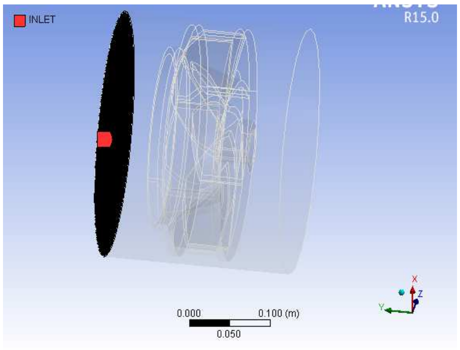

Figure 5: Defining Inlet Duct

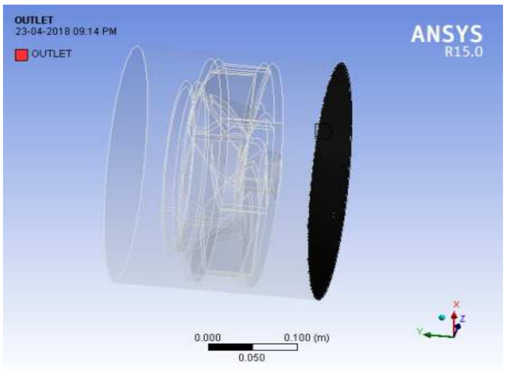

Figure 6: Defining Outlet duct

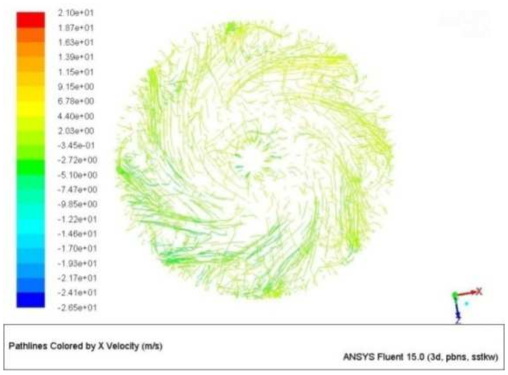

Figure 7: Flow Simulation

RESULTS

Pressure and velocity distribution are shown in the below figures. The velocity distribution is almost uniform unlike heavily concentrated at the eye of the impeller, but due to losses, the velocity has been decreased at the rear ends of the blades. Pressure distribution is also continuous, holding low pressure at the hub and high pressure at the tips. The difference between the suction pressure and the saturation pressure of the fluid is being pumped; it is used to measure how the fluid is to saturated condition. If npsha(available) is maintained above the level than the npsh required by the pump, cavitation can be avoided. In order to prevent cavitation damage, the pump will have a requirement of npsha, failing may cause severe damage. The fluid which has a certain suction pressure will experience losses as it travels from tank to pump through inlet of the pump. Frictional losses and other minor losses in piping are proportional to the square of the flow velocity.

In Steady State

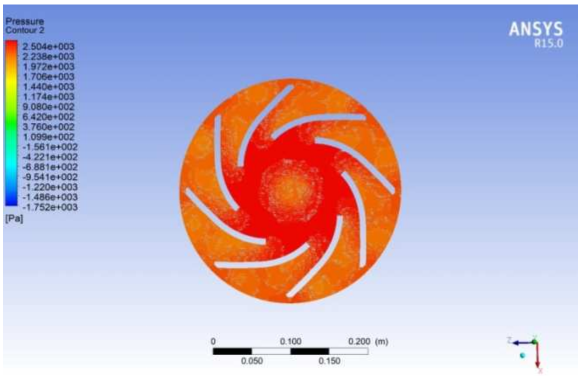

Figure 8: Pressure Contour in Steady State

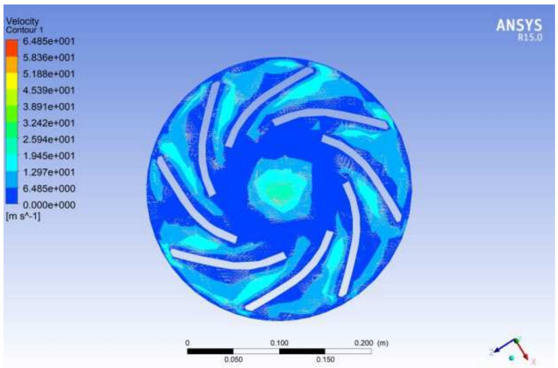

Figure 9: Velocity Contour in Steady State

In Transient State

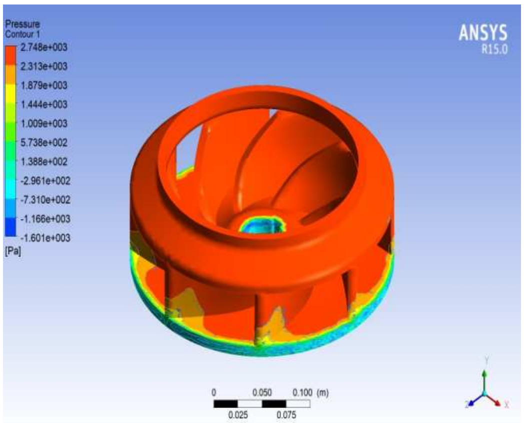

Figure 10: Pressure Contour in Transient State

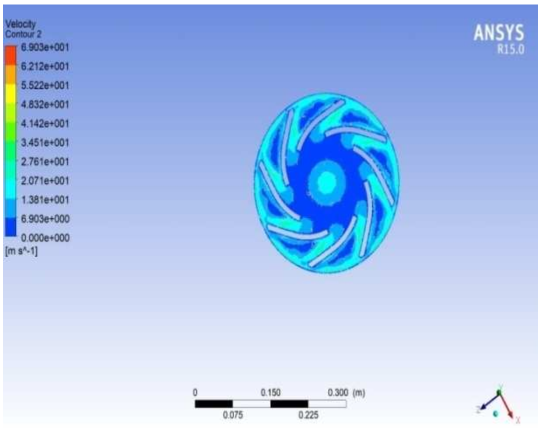

Figure 11: Velocity Contour in Transient State

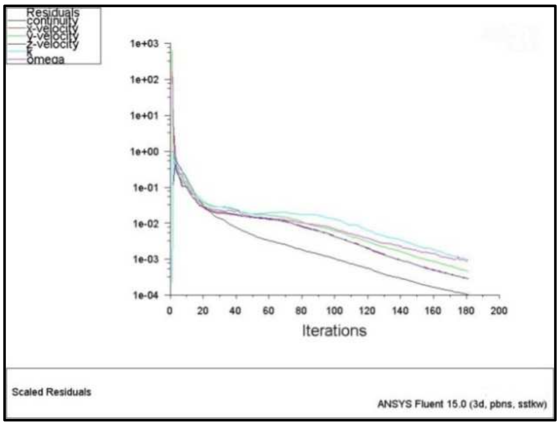

Figure 12: Convergence of Iterations Graph in Steady State

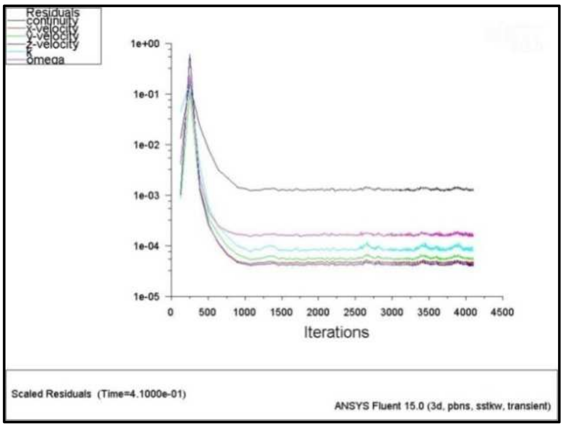

Figure 13: Convergence of Iterations Graph in Transient State

In steady state, the solution is being converged in few number of iterations, but in transient state, the solution is only been stabilized after there is a steep fall in few iterations, and further increased the iterations to 4000 to examine the behavior and to observe any change occurring.

Table 1: (i) Impeller Grid Independent Study (ii) Impeller Study

	Maximum	Minimum
Density	1.225	1.225
Pressure	2878	-680.9
Turbulence kinetic energy	143.1	0.1023
Velocity	64.85	0
Velocity u	-20.33	19.99
Velocity in stn frame u	-24.8	-24.8
Velocity in stn frame v	12.8	-64.78
Velocity in stn frame w	20.69	-22.99
Wall shear stress	11.79	0.041146
Wall shear stress in X	1.293	-1.82
Wall shear stress in Y	0.2473	-11.78
Wall shear stress in Z	2.663	-11.56
Wall shear stress in Y+	2039	9.692
	Maximum	**Minimum**
Density	1.225	1.225
Pressure	2504	-1752
Turbulence kinetic energy	69.29	0.01583
Velocity	66.45	0
Velocity u	32.18	33.29
Velocity in stn frame u	68.53	0
Velocity in stn frame v	384.9	3.9546
Velocity in stn frame w	-33.29	30.2
Wall shear stress	-68.51	11.7
Wall shear stress in X	-31.79	30.48
Wall shear stress in Y	-3.849	3.947
Wall shear stress in Z	-1.02	3.447
Wall shear stress in Y+	-3.496	3.961

CONCLUSIONS

From the results, it is clear that the flow distribution inside the impeller is uniform, unlike only concentrated at the hub. The effect of the cavitation is also minimal; an alteration in the blade angles and geometry of the impeller has a significant influence on the performance. To avoid cavitation, one must ensure that the local pressure at all the points inside the centrifugal pump starts above the saturation pressure of the fluid, so the quantity that is used to determine whether the local pressure inside the pump is adequate with respect to vapor pressure of the flowing fluid, without causing cavitation. The efficiency of the pump increased due to the configuration of the blades. The increase in pressure at any point is proportional to the square of angular velocity and the distance of the point from the axis of rotation. The pressure and the velocity of the fluid are relatively high if the impeller speed is high. The pump works efficiently for non-viscous fluids. It cannot be operated at high heads, if the medium of fluid employed is viscous, it become cumbersome or ineffective. The working of the pump is simple; the pump works with less power consumption.

REFERENCES

1. *Maxime Binama, Alex Muhirwa, Emmanuel Bisengimana “Cavitation Effects in Centrifugal pumps – A Review” Int. Journal of engineering research and applications ISSN: 2248-9622, Vol. 6, Issue 5, (part-1) May 2016, pp52-63.*
2. *Dalia mohamed sadek El-Gazzar said Abdel-Aleem farag hawash “Optimizing centrifugal pump performance by different blade configurations” American journal of mechanical and industrial engineering. Vol3. No. 1, 2018.*
3. *Dr. R. K. Bansal (1998) “A textbook of fluid mechanics and hydraulic machine” 9th edition.*

4. *S. A. I Bellary, A. Hussain, A. Samad and R. A. Kanai "Performance optimization of centrifugal pump for crude oil delivery" The Journal of Engineering Resarch (TJER), Vol. 15. No. 1(2018)88-101.*

5. *Fox, Robert w., Alan T. McDonald, and Philip J. Pritched. "Introduction to fluid mechanics. Vol. 7, (New York: Jhon Wiley & sons, 1985".*

6. *NGH, Vijayapratap R., M. J. Zinzuvadaia, and Saurin M. Sheth. "Parametric study and Design Optimization of Centrifugal pump Impeller-A Review. " Int. Journal of engineering research and Applications (2014): 2248-9622.*

7. *Johann Friedrich Gulich, "centrifugal pumps", second edition, Springer- Verlag Berlin Heidelberg 2008, 2010, Springer Heidelberg Dordrecht London, New york, ISBN 978-3-642-12823-3.*

8. *Pranit M. Patil and R. G. Todkar, "An overview of effects of plitter blades on centrifugal pump performance", International Journal of Engineering Research & Technology, vol.2, issue 11, ISSN 2278-0181.*

9. *Zhang Jinfeng, "Influence of splitter blades on the total flow field of a Low Specific Centrifugal pump" Technology and Research Center of Fluid Machinery Engineering and Technology (IRJET), volume:02 issue: 02 May 2015.*

10. *Reddy, P. R., & Saikiran, M. (2016). Aerodynamic Analysis of Return Channel Vanes in Centrifugal Compressors. International Journal of Mechanical Engineering (IJME), 5(1), 73-82.*

11. *Patel M. G, and A. v. Doshi "effects of impeller blade exit angle on the Performance of Centrifugal pump" Int J. Emerging Technology and Advanced Engineering 3, no.1 (2013): 9I-99.*